Available online at www.sciencedirect.com
ScienceDirect
Procedia Engineering 101 (2015) 135 – 142
Procedia Engineering
www.elsevier.com/locate/procedia

3rd International Conference on Material and Component Performance under Variable Amplitude Loading, VAL2015

Effect of mechanical properties of fasteners on stress state and fatigue behaviour of aircraft structures as determined by damage tolerance analyses

Radek Doubrava*

VZLU (Aerospace Research and Test Establishment), Prague 19905, Czech Republic

Abstract

Fatigue is one of the most common failure modes of structures and components. The prediction of fatigue crack propagation in real structures must take into account realistic boundary conditions. The purpose of this paper is to describe a technique for modelling fasteners through FEM global structure modelling and using the model for stress state analyses and crack growth prediction. The consideration of fastener stiffness during stress analysis is an integral part of the damage tolerance philosophy. The FASTRAN retardation model with a load sequence simulating the real operational service of an aircraft structure was used for crack growth prediction.

Peer-review under responsibility of the Czech Society for Mechanics

Keywords: FEM; FASTRAN; crack growth

1. Introduction

Typical fastener elements used in the aircraft industry include rivets and bolts. The stiffness and strain characteristics of fastener elements can be obtained as follows:

* Corresponding author. Tel.: +420-225-115-134; fax: +420-283-920-018.
E-mail address: doubrava@vzlu.cz

1877-7058
Peer-review under responsibility of the Czech Society for Mechanics
doi:10.1016/j.proeng.2015.02.018

- By testing specimens that are geometrically, mechanically, and technologically similar to real aircraft structures
- By performing numerical simulation with contact analysis
- By finding analytical solutions

One of the methods used for measuring fastener properties is testing a single lap shear joint.

Figure 1 compares the experimental and numerical simulation results for a single lap shear specimen composed of metal and composite components connected by a mechanical joint.

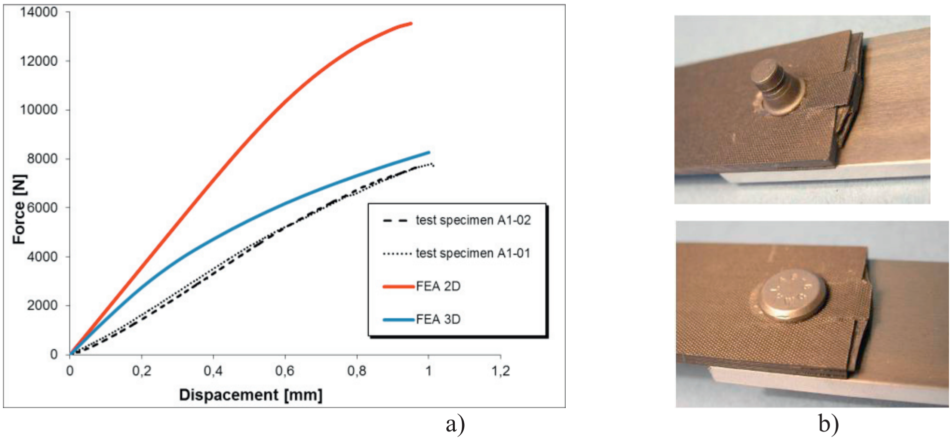


Fig. 1: Comparison of experimental and numerical simulation results obtained using detailed contact model of fastener composed of 2D and 3D elements (a) [2]. Typical damage of composite part when mechanical joint connects metal and composite materials (b) [1].

A detailed FE model can be used to assess fastener stiffness. For a composite material in particular, it is important to use a 3D modelling technique, as illustrated in Figure 1. However, from the point of view of global structure modelling, this technique is problematic in terms of the number of elements and the computational time required.

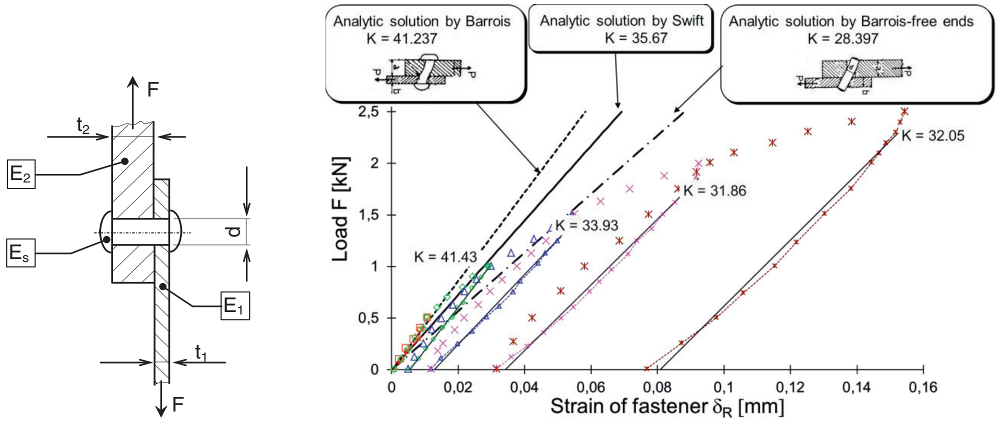
2. Fastener stiffness

Many analytical solutions for determining fastener stiffness were reported by literature [2]. A general form of the solutions is shown below:

$$\delta_R=\delta_R\,(F,\, d,\, t_1,\, t_2,\, E_1,\, E_2,\, E_s) \tag{1}$$

These solutions are based on the geometrical (t = thickness, d = diameter) and material (E = Young's modulus) characteristics of joint elements and fastener (R means rivet).

Figure 2a shows the geometry of a single shear lap rivet joint. Figure 2b compares the analytical solutions [3] and experimental measurements for a typical rivet joint of aircraft wing. Non-linear behaviour and the effect of load history over a 1-kN load are shown in Figure 2b. The value of fastener stiffness was used for analyses of displacement of connected parts and application for numerical simulation.



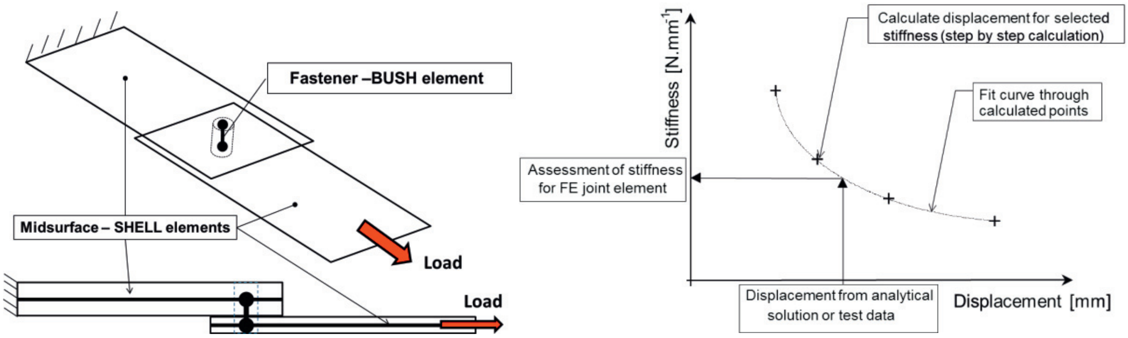
a) b)

Fig. 2: Geometry of a single shear lap rivet joint (a) and comparison between test data (points) and analytical solutions (b) [3].

3. Proposed technique for modelling of fastener

In general, an FE model is limited by the density of the finite element mesh employed, which must be fine near a crack tip and coarse in the other parts of the model. This limitation is due to the computational time required to generate a solution because the geometry of the crack tip changes during crack propagation. From this point of view, models based on the spring element BUSH of the commercial FE codes have been proposed. Because the stiffness of elements of this type of model depends on the local mesh density, a technique based on stiffness assessment using an FE model of a single shear rivet joint is designed for cases in which an analytical or experimental displacement parameter is known. For accurate modelling, it is important for the mesh densities of the global FE model and the FE model used for stiffness assessment to be equal. The proposed model, boundary conditions, and technique used for stiffness assessment are illustrated in Figure 3.

This technique can be used to assess the load versus displacement characteristics for a displacement parameter that exhibits non-linear behaviour because this type of element is also considered a non-linear spring.



a) b)

Fig. 3: Geometry, boundary conditions, (a) and schematic of technique (b) used to assess equivalent stiffness of fastener element.

4. Application to real aircraft structure

The proposed technique is suitable for detailed analyses of large aircraft structures, especially, for damage tolerance analysis by next-generation FE model modifications.

Figures 4 show the application of damage tolerance assessment to a lower wing skin connection. The proposed technique of stiffness rivet modelling was used to determine the rivet load distribution between the skin and other parts such us stringers, flange plates, and overlaps.

A part of the wing was modelled for the following purposes:

- To propose and verify the design of the test specimen for fatigue testing.
- To assess critical points and optimize the design.
- To calculate the fracture mechanical properties of and thereby predict fatigue crack growth and determine the time required for inspection

An FE model was prepared using the FEMAP [4] code for analysis using the NASTRAN [5] FE program. 2D shell elements with isotropic material properties were used. The fasteners were modelled by the technique described in Section 3 using BUSH elements with stiffness tuning of the local mesh size. The local mesh size was approximately 5 mm.

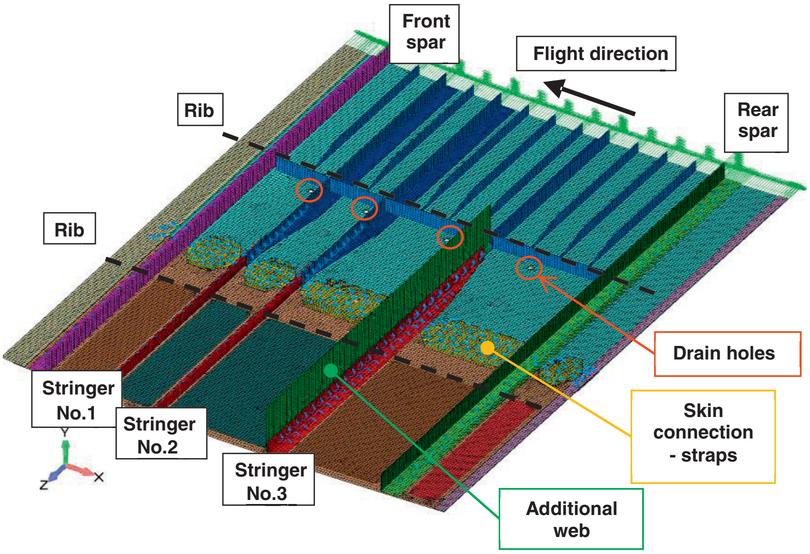


Fig. 4: Proposed FE model of lower skin panel

An analytical method was used for fastener stiffness assessment. Figure 5 compares the analytical models for typical connection dimensions (fastener diameter and thickness of the connecting part).

The analytical method proposed by Swift [2] was implemented to assess the fastener stiffness and equivalent stiffness using the proposed global FE model. The global FE model was constrained according to the boundary conditions of the wing, i.e., symmetry in the XY plane and fixing translation in the Y direction along the position of the ribs and spar web. For loading, a unit stress of 1 MPa was applied at the centre of the specimen by providing displacement in the Z direction along the specimen cross section. Figure 6a shows the numerical simulation results on the global FE model. Based on the calculated load of the rivets and stress analyses, the critical area required for crack growth was determined. Figure 6b shows the results of stress and fastener loading on the global FE model.

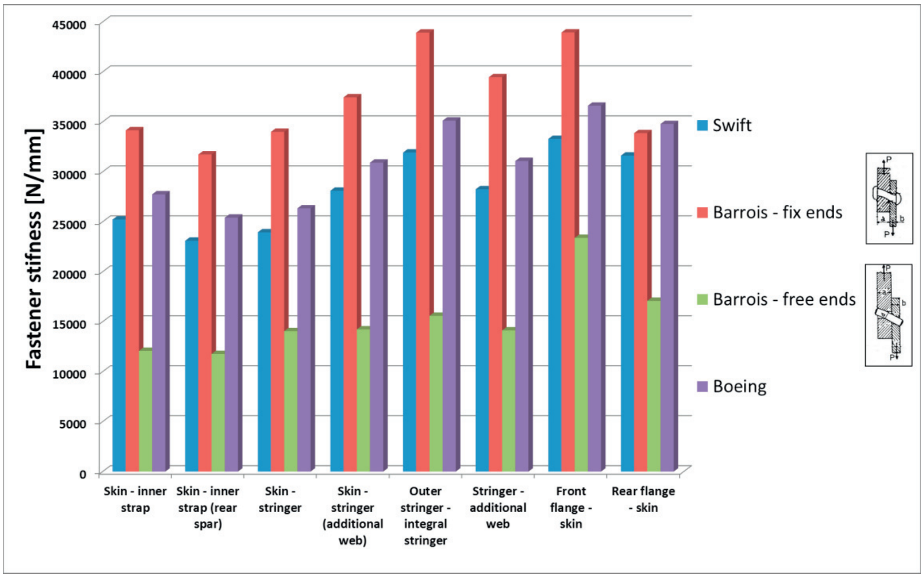
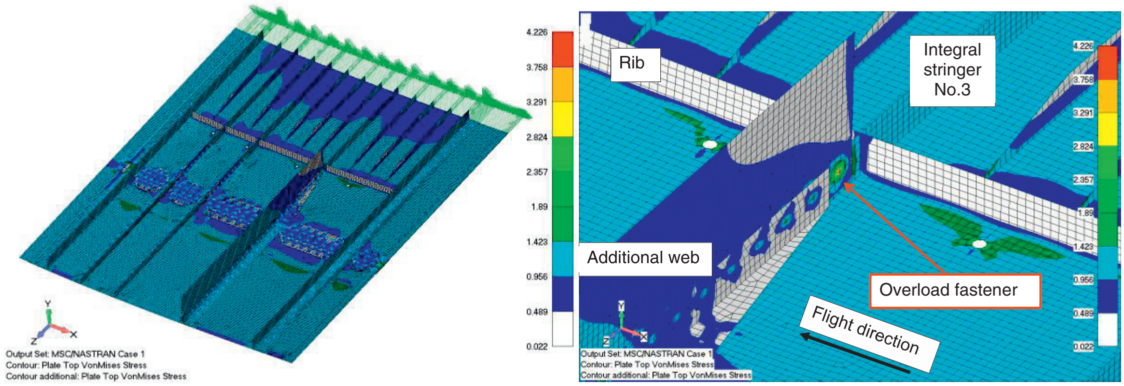


Fig. 5: Comparison of results of analytical methods for fastener stiffness assessment [2]



a) b)

Fig. 6: Contour map of stress analyses in megapascals (a) for proposed FE model and location of critical area for assessment of crack growth prediction (b)

CRAC2D elements from commercial NASTRAN FE code were used to assess the stress intensity factor in the critical area. The local mesh density was increased to predict the crack growth. The VZLU macrocode built in the FEMAP application programming interface [6] was used for the application of CRAC2D elements [7].

The scenario of probable crack growth propagation is based on the recommendations of JSSG 2006 [8]:

1. The primary crack starts from the fastener hole (T1) in integral stringer No. 3 (corner crack, R = 1.3 mm), including the load from the fastener.

2. The secondary crack (T2) propagates in integral stringer No. 3 (starting from corner crack, R = 0.13 mm) – independent increase in the length of the crack for time or cycles calculated for the opposite primary crack. FE calculation starts from the through crack whose length corresponds to the thickness of the stringer.

3. The through crack of the secondary crack (T2) continues to grow on the skin after damage of stringer No. 3 (presumption of through symmetry crack growth from dimension 2a = 6.3 mm).

Figure 7 shows the proposed scenario of crack growth from a fastener hole in the critical area.

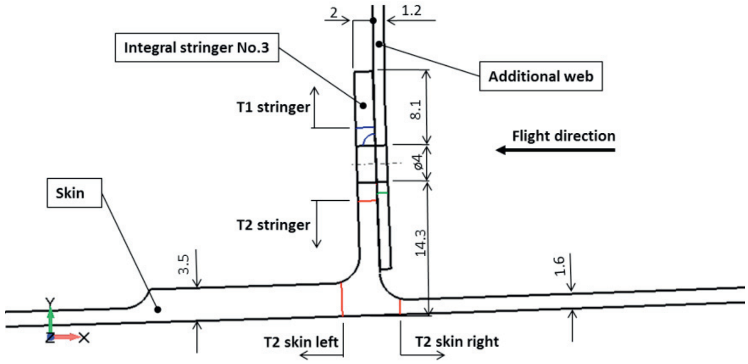


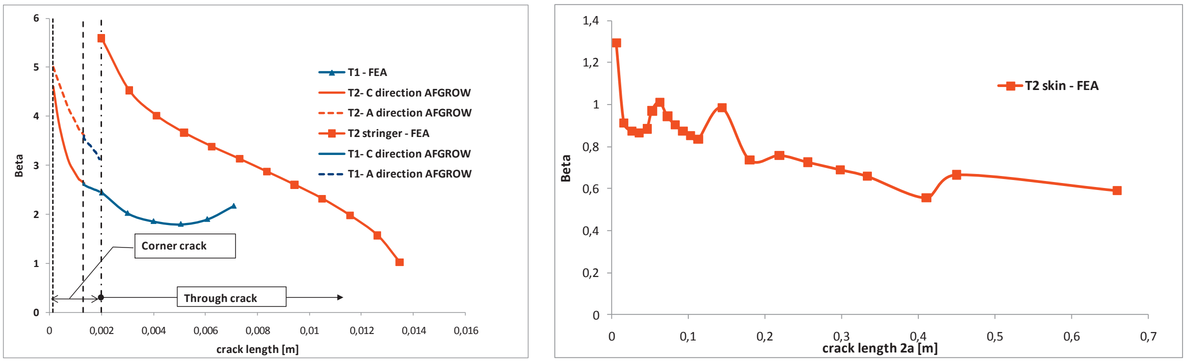
Fig. 7: Cross section of critical area and proposed scenario of fatigue crack growth propagation (all dimensions are in millimetres)

5. Stress state analyses

The value of the stress intensity factor obtained from FE analyses was recalculated using the beta shape function in AFGROW [9] via the following relation:

$$\beta = K_I / \sigma\sqrt{\pi a} \tag{2}$$

Figure 8 shows the beta function calculated for the critical area. For studying corner crack growth propagation, an analytical function that includes a crack growth prediction code such as AFGROW [9] can be used. The fastener load determined using the proposed technique is an important input to be considered for the bearing load in the analytical model.



a) b)

Fig. 8: Beta function determined from combination of analytical functions and results from global FE model at start of crack propagation from fastener hole (a) and propagation of geometry symmetry (left and right direction) through crack on skin (b)

6. Material model for crack growth prediction

The integral lower skin panel is made from 7475-T7351 material. On the basis of reference [10], the FASTRAN retardation model was used for the assessment of fatigue crack growth prediction.

The input material parameters for the FASTRAN retardation model are as follows:

– Static strength (ultimate and yield stresses)
– Crack growth rate as a function of the effective stress intensity factor measured under constant amplitude loading and supplemented by data from the presented fractographic analysis
– Fracture toughness

On the basis of the test analysis, the Paris law coefficient of the 7475-T7351 material shaped in the form of a 76-mm-thick plate was determined [11]:

$$da/dN = 1.92321 * 10^{-10} * (K_{eff})^{3.134} \tag{3}$$

A previous paper [11] reported the calculated parameters of crack propagation in a 76-mm-thick plate made of 7475-T7351 material loaded in the longitudinal direction (LT) for use in the FASTRAN crack retardation model of AFGROW [9]. The characteristics of the transition zone between the tensile and shear modes of the fatigue crack are determined on the basis of fractographic analysis of the fracture surfaces of the specimens used for the measurement of the speed of crack propagation. Typical speeds for the transition zone boundary mode violations range from 5.3819e-7 m/wave (beginning zone) to 2.001728e-5 m/wave (end zone) [11].

For understanding the influence of the retardation model, comparative analyses of crack growth prediction with and without retardation influence were performed.

7. Load sequence

The load sequence was designed to simulate the real operational service of an aircraft (Figure 9). Only one parametric spectrum was available, so the test sequence was designed in block form for each phase of a typical flight: take off; flight; and landing, including the landing impact, wheel rebound, braking, cornering, and the phase after landing on the ground with the landing weight. A previous paper [12] presents the build sequence for simulating the full-load operating conditions of a small transport aircraft, which is assembled with an estimated life of 30,000 operating hours. One repeat sequence covers 3000 flight hours. Figure 15 shows the generated load sequence using AFGROW for the assessment of fatigue crack growth prediction.

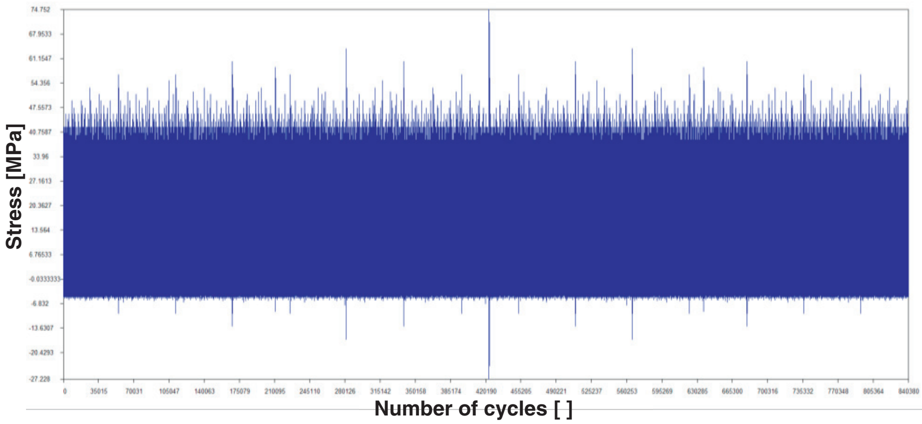
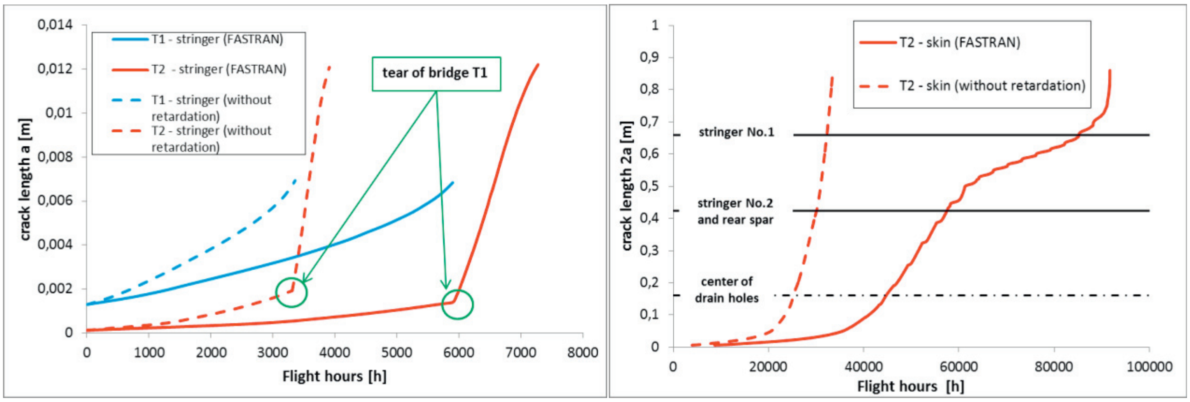


Fig. 9: Load sequence in block form used for crack growth prediction [12]

8. Results of fatigue crack growth prediction

The AFGROW [9] code was used for crack growth prediction. The fastener load determined using the proposed technique is an important input to be considered for the bearing load in the analytical model.

Figure 10 shows the results of fatigue crack growth prediction in the lower skin panel. The results are presented in terms of the growth curve.



a) b)

Fig. 10: Crack growth curve for initial phase of growth from rivet hole in integral stringer (a) and for crack growth in skin (b) (dashed curve = no retardation model, full curve = FASTRAN retardation model). Crack labelling is shown in Figure 7.

The results in Figure 10 show the important effects of applying retardation in fatigue crack growth prediction. The effects of retardation confirm the sensitivity of this model to real structure geometry changes, e.g., increase in crack speed near the drain hole or decrease in crack speed before the integral stringer. The effect of retardation is significant in long cracks.

9. Conclusion

This paper describes a technique for modelling fasteners; the technique uses an FE model for predicting fatigue crack growth propagation. This technique considers the significant effect of fastener stiffness on the prediction of fatigue crack growth. The inclusion of the fastener and the fastener stiffness in predicting the critical area required for crack growth is important under the effects of a bearing load and the surrounding structure.

The application of the FASTRAN retardation model, which was verified by testing [10], resulted in improved assessment of the fatigue behaviour and detection of important inputs for determining service inspection intervals.

Acknowledgments

This study received financial support through the research organisation development program of the Ministry of Industry and Trade of the Czech Republic.

References

[1] Jironč, J.: Research of static load capacity mechanical and bonded joints in combination of metal and composite structures. Internal report VZLU R-4086, 2007
[2] Skorupa, A., Skorupa, M.: Riveted Lap Joints in Aircraft Fuselage - Design, Analysis and Properties, Springer Science+Business Media Dordrecht, ISBN 978-94-007-4282-6 (eBook), 2012
[3] Doubrava, R.: Toughness characteristics for modelling of equivalent stiffness rivet connection by FEM, Internal report VZLU R-2936/98
[4] Femap - Finite Element Modeling and Postprocessing, Version 11.1.2. Siemens Product Lifecycle Management Software Inc., 2014
[5] Nastran - NASA STRucture ANalysis. Finite element analysis code. Version NX/Nastran 8.5. Siemens Product Lifecycle Management Software Inc., 2013
[6] FEMAP API Programing Manual, 2011
[7] Mengdehl, C.: CRACK macro for FEMAP/MSC.NASTRAN, Internal report VZLU R-5109, 2011
[8] JSSG-2006, Department of Defense Joint Service Guide: Aircraft Structures, 1998
[9] Harter, J.A.: AFGROW Users Guide and Technical Manual, AFRL-VA-WP-TR-2008, 2008, Air Force Research Laboratory, Ohio, USA
[10] Běhal, J., Nováková, L.: Stress state factor evaluation based on a fractographic analysis for use in the crack growth FASTRAN retardation model of the AFGROW computing code, Engineering Failure Analysis 35 (2013) p. 645–651, 2013
[11] Běhal, J.: Fatigue crack growth computational data of 7475-T7351 alloy in the plate 76-mm thickness of ALCOA production for application in FASTRAN retardation model of AFGROW code, MOSTA.0438.V.U.PD, VZLU 2012
[12] Běhal, J.: Load sequence of L410NG, MOSTA.0429.V.U.PD, VZLU 2012